REPORT DOCUMENTATION PAGE

AFRL-SR-AR-TR-04-

0225

1. REPORT DATE *(DD-MM-YYYY)*

2. REPORT TYPE Final Report

3. DATES COVERED *(From - To)* Mar 1, 03 - Dec 31, 03

4. TITLE AND SUBTITLE
Simulation of Moving Boundary Flow Using Adaptive Cartesian/Prism Grids and DES

5a. CONTRACT NUMBER

5b. GRANT NUMBER F49620-03-1-0202

5c. PROGRAM ELEMENT NUMBER

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER

6. AUTHOR(S)
Dr. Z. J. Wang
Dr. Laiping Zhang

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Michigan State University
2555 Engineering Building
East Lansing, MI 48824

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Department of the Air Force
Air Force Office of Scientific Research
4015 Wilson Blvd.
Arlington, VA 22203-1954

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT

Distribution Statement: Approved for public release; distribution unlimited

13. SUPPLEMENTARY NOTES
DODAAD CODE:
Program Manager: Dr. Fariba Fahroo

20040426 067

14. ABSTRACT

Development of improved weapon systems requires better understanding of the complex aerodynamics created by moving boundaries. Carriage and release of conventional weapons from aircraft and aerial refueling fall in this category. The flow problems involving moving boundaries are very challenging to compute because the resultant flow field is intrinsically unsteady. Modeling such flow problems is made even more difficult due to the complex geometries and the highly turbulent nature of the unsteady flows. Under an exploratory grant from the AEOSR, a new simulation environment for moving boundary problems is being developed. This environment is designed to advance the state-of-the-art for moving boundary flow problems in the following areas: I. Improve the solution accuracy and efficiency through the use of overset adaptive Cartesian/prism grids. Key flow features will be automatically captured with solution based grid adaptations; 2. Improve the accuracy for highly turbulent unsteady separated flows encountered in moving boundary problems with a hybrid RANS/LES (Reynolds Averaged Navier Stokes/Large Eddy Simulation) approach named Detached Eddy Simulation (DES).

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			Dr. Z. J. Wang
					19b. TELEPHONE NUMBER *(Include area code)* 517-432-9132

Simulation of Moving Boundary Flow Using Overset Adaptive Cartesian/Prism Grids and DES

By

Z.J. Wang and Laiping Zhang
Department of Mechanical Engineering
Michigan State University
2555 Engineering Building
East Lansing, MI 48824
Tel: 517-432-9132
Fax: 517-353-1750

September 2003

Final ~~Annual~~ Performance Report
For the Period of Mach 1, 2003 – September 30, 2003
Air Force Office of Scientific Research
4015 Wilson Boulevard, Room 713
Arlington, VA 22203-1954

Contract Number: F49620-03-1-0202
Program Director: Major William Hilbun/Dr. Fariba Fahroo

SUMMARY

Development of improved weapon systems requires better understanding of the complex aerodynamics created by moving boundaries. Carriage and release of conventional weapons from aircraft and aerial refueling fall in this category. The flow problems involving moving boundaries are very challenging to compute because the resultant flow field is intrinsically unsteady. Modeling such flow problems is made even more difficult due to the complex geometries and the highly turbulent nature of the unsteady flows.

Under an exploratory grant from the AFOSR, a new simulation environment for moving boundary problems is being developed. This environment is designed to advance the state-of-the-art for moving boundary flow problems in the following areas:

1. Improve the solution accuracy and efficiency through the use of overset adaptive Cartesian/prism grids. Key flow features will be automatically captured with solution based grid adaptations;
2. Improve the accuracy for highly turbulent unsteady separated flows encountered in moving boundary problems with a hybrid RANS/LES (Reynolds Averaged Navier Stokes/Large Eddy Simulation) approach named Detached Eddy Simulation (DES).

Significant progresses have been made in the following areas:

1. The development of an adaptive Cartesian grid generator and hole cutter named OverCart is near completion. The grid generation part has been concluded, and progresses are being made in the hole-cutting part;
2. A DES (Detached Eddy Simulation) approach has been implement into the flow solver named MUSIC (MUlti-physics SImulation Code), and tested for several validation cases.

The report documents the research activities carried out since the grant award, and outlines future work.

1. INTRODUCTION

1.1 Background of Proposed Research and Program Objective

Development of improved weapon systems requires better understanding of the complex aerodynamics created by moving boundaries. Carriage and release of conventional weapons from aircraft, aerial refueling, and formation flying of two or more aircraft fall in this category. The flow problems involving moving boundaries are very challenging to compute because the resultant flow field is intrinsically unsteady. Modeling such flow problems is made even more difficult due to the complex geometries and the highly turbulent nature of the unsteady flows.

The current state of the art for tackling complex unsteady 3D viscous moving boundary flows is the use of structured overset Chimera method. For example a group in NASA Ames Research Center led by Meakin [1-4] successfully simulated the unsteady flow around a rotorcraft in forward flight. The cost associated with such a simulation is, however, still very expensive in terms of both human resources and computer time. Even with grid oversetting, generating an appropriate overset **structured** grid system for a complex configuration is still time consuming. Because of the fixed topology of a structured grid, it cannot compete against an unstructured grid in terms of adaptivity or flexibility. During the last one and half decades, the unstructured grid methodology has clearly demonstrated its flexibility in tackling complex geometries, and in flow-based grid adaptations [5-12]. The success demonstrated by the unstructured grid method for steady flow problems has prompted considerable development for unsteady moving boundary problems. For example, unstructured tetrahedral meshes have been used quite extensively in tackling complex unsteady 3D flows by Lohner et al [5-7]. There are two approaches in handling moving boundaries using unstructured grids, and their pros and cons are summarized next.

<u>Approach One: Grid Deformation</u>: Grid cells close to moving boundaries are deformed locally using spring-analogy type of algorithms while the grid topology or connectivity is preserved. This approach is very efficient, and does not require solution interpolation. However, the moving boundaries cannot make large relative motions, or shear-type motions. In addition, the grids in some regions may become too coarse or too fine.

Approach Two: Hybrid Grid Deformation/Regeneration: This approach was designed to handle large relative motions. Therefore, arbitrary motions can be supported by this approach. However, frequent grid regeneration and solution interpolation are costly, and can also degrade the solution accuracy.

In almost all moving boundary flow simulations carried out so far, the Reynolds-Averaged Navier-Stokes (RANS) approach with turbulence models has been used. These models were calibrated according to turbulent boundary layers. The use of the RANS approach for highly separated flows can cause large errors [13].

The ultimate objective of the effort is to develop a computational tool that significantly improves on the current state-of-the-art methodology for moving-body problems. In this project, we plan to develop an overset adaptive Cartesian/prism grid method for arbitrary moving boundary problems. Prismatic grids around solid boundaries will be generated nearly automatically through surface extrusions. These prism grids are then overlapped with an automatically generated adaptive Cartesian background grid, which is used for the purpose of coupling the prismatic grids. Due to the fully unstructured nature of the adaptive Cartesian grid, the grid resolution of the Cartesian grid can easily match that of the prismatic grids near the outer boundaries of the prismatic grids, therefore eliminating one serious problem associated with overset structured grids, i.e., resolution mismatch near grid interfaces. Holes are then generated automatically in the adaptive Cartesian grid to facilitate data communication between the Cartesian and prismatic grids. The use of prismatic grids can enhance the solution accuracy in the viscous boundary layer [9], and Cartesian grids are much more efficient in filling a given space with a specified length scale than tetrahedral grids [12]. In addition, a promising hybrid RANS/LES approach named Detached Eddy Simulation (DES) [14] will be implemented to improve the solution accuracy for highly separated unsteady flows. Preliminary computations with DES have shown significant improvements in accuracy over RANS [13,15,16]. Therefore, the main advantages of the proposed methodology for moving-body applications are as follows: (i) a significant reduction in the problem set-up time; (ii) a significant reduction in the calculation turn-around time; (iii) accuracy enhancements due to the use of DES.

1.2 Research Objectives

The ultimate objective of the effort is to develop and demonstrate a computational tool that can tackle moving-body problems such as munitions/aircraft separations accurately, efficiently and in an automated fashion. The improvements planed for the first year include the use of overset adaptive Cartesian/prism grids, and the implementation of the Detached Eddy Simulation approach for turbulent unsteady moving body flow problems. The tool will provide better understanding of the complex aerodynamics created by moving boundaries.

More specifically, the following objectives are set for this project:

1. Improve the solution accuracy and efficiency through the use of overset adaptive Cartesian/prism grids.

2. Improve the accuracy for highly turbulent unsteady separated flows encountered in moving boundary problems with Detached Eddy Simulation (DES). The DES approach will be evaluated with documented experimental data.

1.3 Outline of the Report

The next section describes the adaptive Cartesian/prism grid generation method. Issues on data structures and robustness of prism grid generation are discussed. After that, the algorithms for hole cutting to facilitate data communication between overlapped prism and Cartesian grids are presented in Section 3. Then the enhancement to the arbitrary grid finite volume flow solver is outlined in Section 4. Specifically the DES approach is presented in detail. Next, several verification and demonstration cases are presented to test the new development. Finally, conclusions and recommendations for further research activities are included in Section 6.

2. GRID GENERATION AND HOLE CUTTING

1.1 Generation of Adaptive Cartesian Grid and Prism Grid

A graphical user interface (GUI) based adaptive Cartesian/prism grid generator – named OverCart - is being developed to generate overset adaptive Cartesian/prism grids given triangulated surface definitions. The visualization window of OverCart is displayed in Figure 2.1, where generic wing and missile geometries are shown. Once the surface triangulations are given, the surface mesh is "extruded" in the surface normal directions [10] to generate the prism grids around all geometric entities, such as the wing and the missile. To ensure the simplest possible data communication pattern, it is required that the outer boundaries of the prism grids do not intersect each other. If they do, the prism grids are "compressed" locally until there is no intersection. With this requirement, the prism grids never communicate among themselves, but communicate only with the stationary adaptive Cartesian grid. For concave geometries with sharp corners, it is possible that the "extruded" prism cells may intersect each other. Various smoothing techniques are implemented to prevent the intersections.

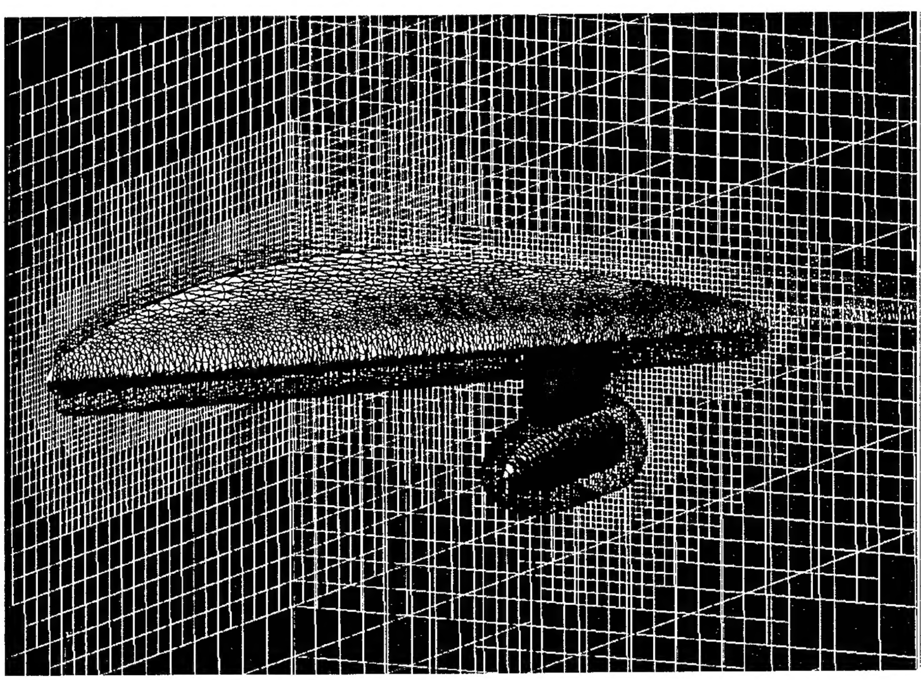


Figure 2.1. Visualization Window of OverCart

After the prism grids are produced, a tree-based adaptive Cartesian grid is generated automatically matching the grid resolution near the outer boundaries of the prismatic grids. In order to support arbitrary local anisotropic grid adaptations, we will adopt the so called 2^N tree data structure [12]. In a 2^N tree, each tree node can have 2, 4 or 8 children nodes, supporting seven possible subdivisions for a Cartesian cell, as shown in Figure 2.2. As a concept demonstration, only the Octree is implemented in OverCart. Note that the popular Octree data structure [20-21] is a special case of the 2^N tree. The 2^N tree is very efficient in capturing flow features aligned with the coordinate directions. Even if a feature is not aligned with a particular coordinate direction, the 2^N tree can still be many times more efficient than the Octree.

To start the grid generation process, the following parameters are given: the size of the computational domain, and the minimum and maximum grid sizes. Then the following steps will be employed to generate the initial grid:

- Generate a single root node (Cartesian cell) based on the domain size;
- Recursively subdivide the root node until all cells are smaller than the specified maximum cell size;
- Identify all Cartesian cells intersecting the outer boundaries of the prismatic grids;
- Recursively refine the intersected cells until all cells intersecting the interfaces match the grid resolution of the prismatic grids;

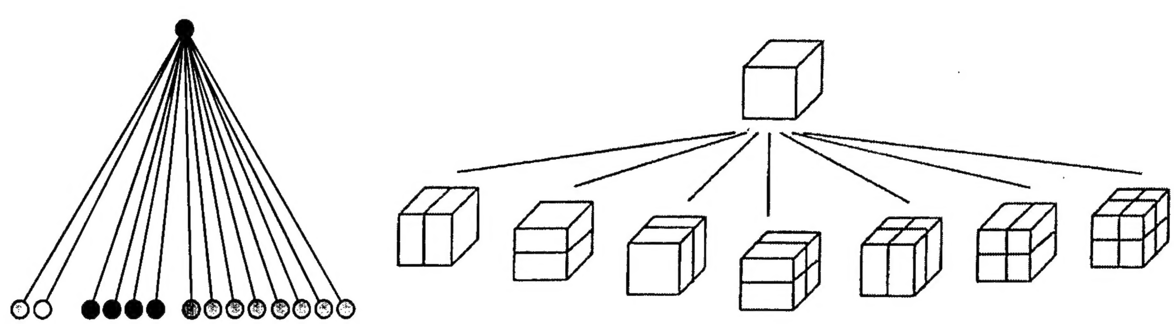


Figure 2.2 Seven Possible Types of Cell Subdivision of a Cartesian Cell with the 2^N Tree

The final adaptive Cartesian grid is smoothed so that the length scales between two neighboring cells do not differ by a factor more than 2 in any coordinate direction. In addition, several buffer layers with the same grid resolution near the outer boundaries of the prismatic grids are used to minimize local discretization error. Once a tree-based grid is generated, it is "flattened" into a list based grid format, i.e., point, face and cell lists and the associated connectivity are explicitly computed. This list-based grid is then fed into a list based flow solver, such as the MUSIC flow solver.

2.2 Automated Hole Cutting, and Donor Cell Identification

The use of overset adaptive Cartesian and prismatic grids has the potential of handling arbitrary moving boundary problems without any user interference. A critical element in achieving this level of automation is an automated hole-cutting algorithm, in which invalid Cartesian grid cells (cells inside a solid body) are excluded from the calculation, and donor cells are identified for hole boundary cells and outer boundary cells (the outer most layer in the prismatic grid). A schematic of the hole-cutting operation with definition of terminology is shown in Figure 2.3. We are now in the process of extending an automated hole-cutting algorithm developed for structured grids to overset adaptive Cartesian and prismatic grids to perform hole-cutting [22]. The efficiency of the hole-cutting algorithm is critical since many steps of the hole-cutting operation will be performed in a moving boundary problem as the prismatic grids move in the flow field. To achieve the maximum efficiency for the hole-cutting algorithm, two search trees will be used extensively. One is the 2^N tree used in generating the adaptive Cartesian grid, and the other is an alternating digital tree (ADT) [23] for the bounding boxes of the prismatic cells. The use of the 2^N tree to speed up search operations is another significant advantage of using the adaptive Cartesian grid for moving boundary flows. The hole-cutting algorithm consists of the following steps:

1 Solid boundaries of the prismatic grids are used to generate hole-boundaries in the adaptive Cartesian grids. Again the 2^N tree will be used to identify Cartesian cells which intersect the solid boundaries;

2 Generate a list of hole boundary cells in the adaptive Cartesian grid, and blank all the Cartesian cells which are inside the solid boundary. Use the ADT tree to find the prismatic cells, which bound the centroids of hole boundary cells. The prismatic cells bounding the centroids of the hole boundary cells are also called donor cells.

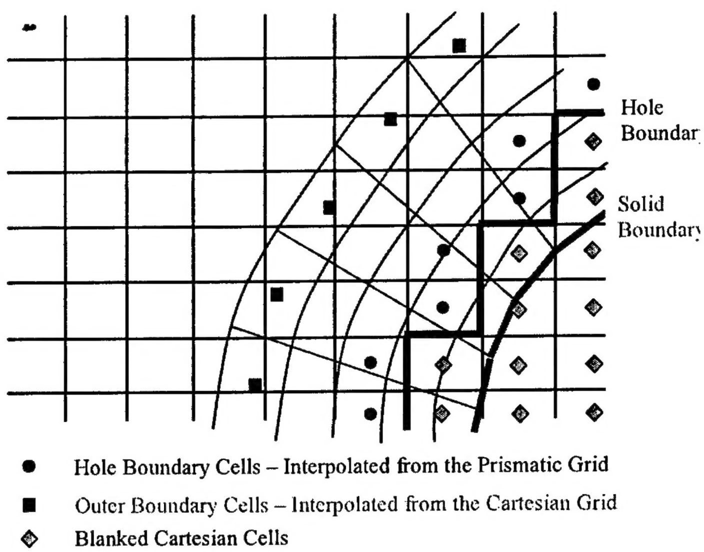


Figure 2.3. Schematic of Hole-Cutting

3 Generate a list of outer boundary cells, and use the 2^N tree (again!) to identify the Cartesian cells which bound the cell-centroids of the outer boundary cells of the prismatic grids. These Cartesian cells are also called donor cells;

After each time step, field variables at the outer boundary cells are interpolated from the Cartesian grid, while solutions at the hole boundary cells are interpolated from the prismatic grids.

3. ENHANCEMENT OF FLOW SOLVER

3.1 Review of Current Capability

The new developments proposed in this project is being incorporated into a highly flexible software platform named MUSIC (MUlti-physics SImulation Code), developed by the PI in the CFD Laboratory of MSU. The design philosophy of MUSCI is to handle *arbitrary* computational grids and *arbitrary* physics in a tightly coupled environment. In order to achieve software modularity, extensibility and reliability, MUSIC is based on the object-oriented programming paradigm, and written in C++. The following requirements have been met in the design of MUSIC:

1. No global variables;
2. Access to and from data performed through well-defined object interface functions;
3. Support multi-zone arbitrary grid topology (including structured, unstructured and adaptive grids);
4. Support any physics, boundary conditions, control parameters and solution variables;
5. Support multi-grid and parallel computing with MPI serving as the message-passing standard;
6. Support moving and dynamic grids;

The flow solver module in MUSIC is based on a second-order finite volume method. The flow solver has the following major features:

1. Cell-centered finite volume method capable of handling arbitrary polyhedral grids. The solver employs a face-based data structure, and is very efficient for arbitrary grids;
2. Use of a least squares linear reconstruction algorithm to ensure second-order accuracy on arbitrary grids;
3. A variety of Riemann solvers including Roe, and AUSM splitting;
4. Use of an implicit block Lower-Upper Symmetric Gauss-Seidel solution algorithm for the non-linear system for fast convergence;

The code is being enhanced to support the current project. For example, the code has been extended to handle arbitrary moving boundary problems. Another major extension is to implement the DES capability outlined below.

3.3 Detached Eddy Simulation

The DES approach was developed by Spalart et al. to overcome the deficiencies of RANS models for predicting massively separated flows [15]. The objective was to develop a numerically feasible and accurate approach by combining the most favorable elements of RANS models and LES. The primary advantage of DES is that it can be applied at high Reynolds numbers as can RANS approaches, but also resolves geometry dependent, unsteady three-dimensional turbulent motions as in LES. Many DES predictions to date [13-16] have been favorable, forming one of the motivations for the present research. The DES proposed by Spalart et al. combines the Spalart-Allmaras model with LES, and its major components are described next.

Spalart-Allmaras Model The Spalart-Allmaras one-equation model [24] (referred to as the S-A model) solves a single partial differential equation for a variable $\tilde{\nu}$ which is related to the turbulent viscosity. The differential equation is derived by using empiricism and arguments of dimensional analysis, Galilean invariance and selected dependence on the molecular viscosity.The model includes a wall destruction term that reduces the turbulent viscosity in the log layer and laminar sublayer. The equation can be written in the following form

$$\frac{D\tilde{\nu}}{Dt} = c_{b1}\tilde{S}\tilde{\nu} - c_{w1}f_w\left[\frac{\tilde{\nu}}{d}\right]^2 + \frac{1}{\sigma}\left[\nabla \bullet ((\nu+\tilde{\nu})\nabla\tilde{\nu}) + c_{b2}(\nabla\tilde{\nu})^2\right]. \tag{1}$$

The turbulent viscosity is determined via,

$$\nu_t = \tilde{\nu} f_{v1}, \quad f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}, \quad \chi \equiv \frac{\tilde{\nu}}{\nu}, \tag{2}$$

where ν is the molecular viscosity. Using S to denote the magnitude of the vorticity, the modified vorticity is defined as

$$\widetilde{S} \equiv S + \frac{\widetilde{\nu}}{\kappa^2 d^2} f_{v2}, \quad f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}, \tag{3}$$

where d is the distance to the closest wall. The wall destruction function is defined as

$$f_w = g\left[\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6}\right]^{1/6}, \quad g = r + c_{w2}(r^6 - r), \quad r \equiv \frac{\widetilde{\nu}}{\widetilde{S}\kappa^2 d^2}. \tag{4}$$

The closure coefficients are given by:

$$c_{b1} = 0.1355, \sigma = 2/3, c_{b2} = 0.622, \kappa = 0.41, c_{w1} = \frac{c_{b1}}{\kappa^2} + \frac{1 + c_{b2}}{\sigma}, c_{w2} = 0.3, c_{w3} = 2, c_{v1} = 7.1$$

Detached Eddy Simulation A new length scale is defined for DES, i.e.,

$$\widetilde{d} = \min(d, C_{DES}\Delta),$$

where C_{DES} is a constant, and Δ is the measure of local mesh spacing, taken to be the maximum distance from the current cell centroid to the centroids of its neighbors. Then the distance to the closest wall d in the *S-A* model is replaced with the new length scale $\widetilde{d}$ to obtain DES. The purpose of using this new length is that in boundary layers, Δ far exceeds d and the standard S-A model rules since $\widetilde{d} = d$. The model comes with its experience base and fair accuracy. Away from walls, we have $\widetilde{d} = C_{DES}\Delta$ and the model turns into a simple one equation Sub-Grid-Scale (SGS) model, close to Smagorinsky's in the sense that both make the "mixing length" proportional to Δ. We are counting on the weak sensitivity of LES to its SGS model away from walls, and have only C_{DES} to adjust. On the other hand, the approach retains the full sensitivity to

the RANS model's predictions of boundary separation. The constant was set at 0.65 through the study of homogeneous turbulence [13].

4. VERIFICATION AND VALIDATION CASES

4.1 Turbulent Boundary Layer

This case is selected to validate the newly implemented Spalart-Allmaras turbulence model. Although the physical problem is two-dimensional, it was run as a three dimensional one with periodic boundary conditions in the span-wise direction. The computational grid is displayed in Figure 4.1, and has 100 cells in the stream-wise direction, 40 cells in the wall normal direction and 4 cells in the spanwise direction. The minimum grid size near the wall has a y+ value of 1. The simulation was performed for 400 iterations with an implicit block LU-SGS scheme. The convergence history is displayed in Figure 4.2. The density residual was reduced by 7 orders within 400 iterations. The velocity profile is compared with the law-of-the-wall in Figure 4.3. Note that the agreement is good, indicating that the implementation of the Spalart-Allmaras model was successful.

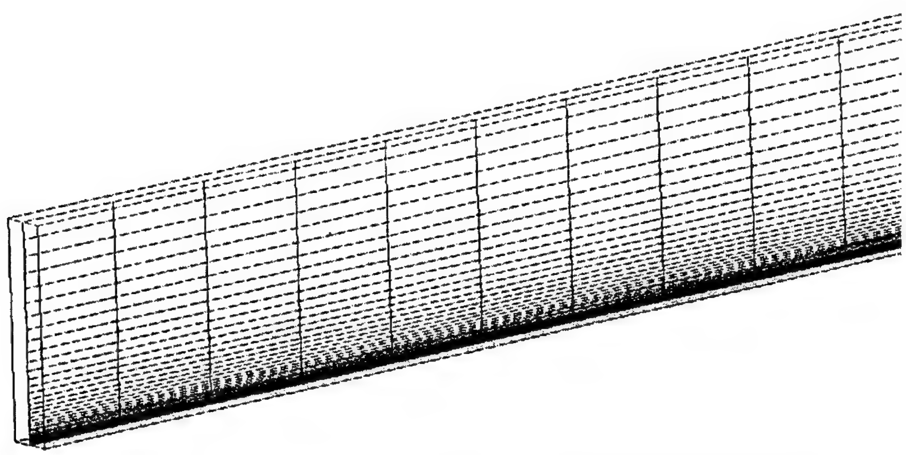


Figure 4.1 Computational grid for a turbulent boundary layer

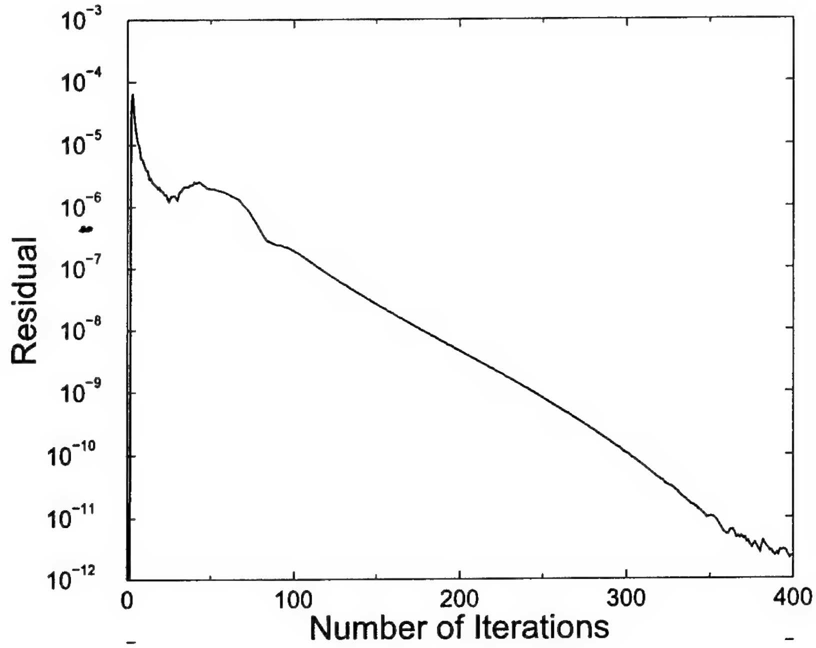


Figure 4.2 Convergence history for the turbulent boundary case

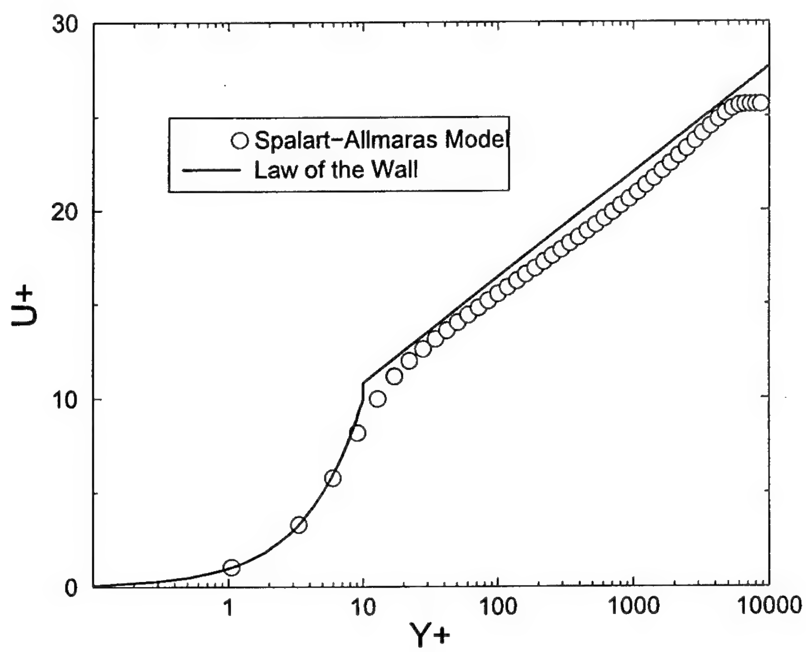


Figure 4.3 Comparison of the velocity profile with the law of the wall

4.2 Turbulent Flow over Oscillating NACA0012 Airfoil

The second-case is to verify the moving boundary capability of the flow solver. The geometry is the NACA0012 airfoil, and the airfoil oscillates around its quarter chord. The flow parameters are: $\alpha_m = 4.86°$, $\Delta\alpha = 2.44°$, $\kappa = 0.0810$, $M_\infty = 0.6$, Re = $4.8\text{x}10^6$. The unsteady moving body simulation started from the steady-state solution. Then 50 time steps are used for each cycle to capture the unsteady features. Figure 4.4 displays the computed Mach number contours at several different angles of attack. For comparison purposes, an inviscid simulation was also carried out with the same flow conditions. The computed Mach contours from this inviscid simulation are shown in Figure 4.4b for the same angles of attack. Note that the flow fields are quite different near the airfoil, especially at the trailing edge. The turbulent boundary layer and the wake are visible in the viscous solutions. The plot of the lift coefficient vs. the angle of attack is shown in Figure 4.5. Results from both the inviscid and viscous simulations are plotted in the figure. It is obvious that the viscous simulation agrees much better with the experimental data than the inviscid calculation at all angles of attack.

In order to verify that the solution is grid-independent, a grid refinement study was also carried out. Two finer grids with about 30K and 55K cells are generated, and used in the same simulation. Figure 4.6a shows the plots of lift coefficient vs. the angle of attack computed from the viscous simulations using these three different grids. The minimum and maximum C_L on the three grids are (0.4114, 0.9342), (0.4081, 0.9275), (0.4051, 0.9230) respectively. The differences in C_L between the coarsest and finest meshes are less than 2%. Figure 4.6b displays the corresponding plots from the inviscid simulations using two different grids with 8.2K and 16.4K cells respectively. Grid independence is demonstrated for both the inviscid and viscous cases.

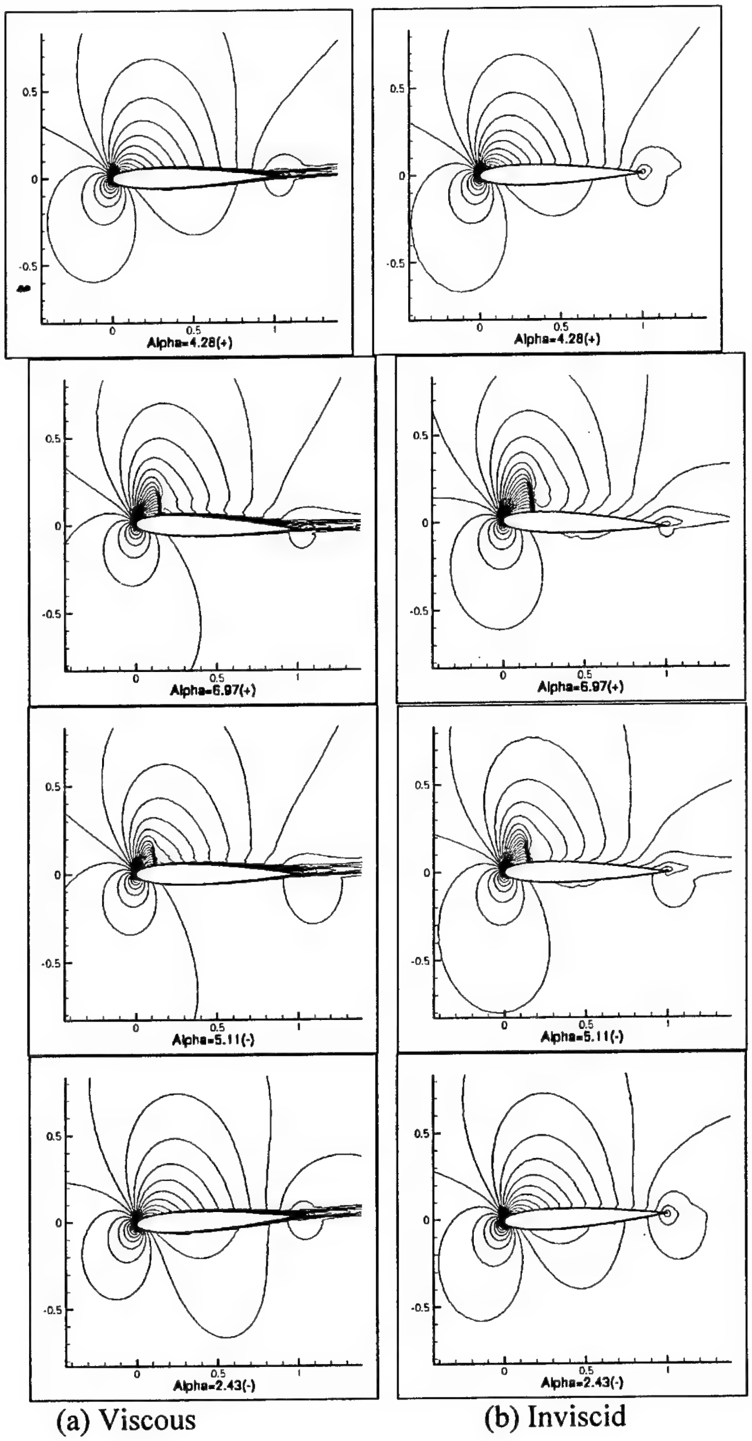


Figure 4.4. Computed Mach number contours at different times

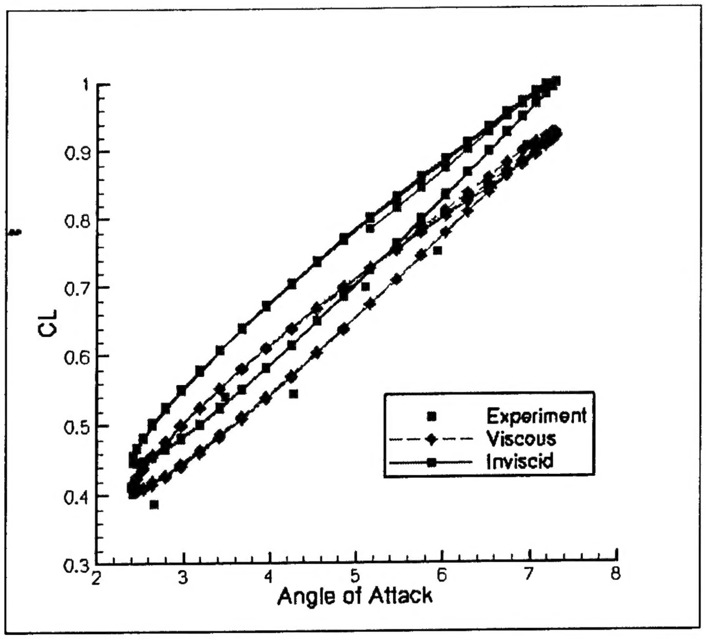
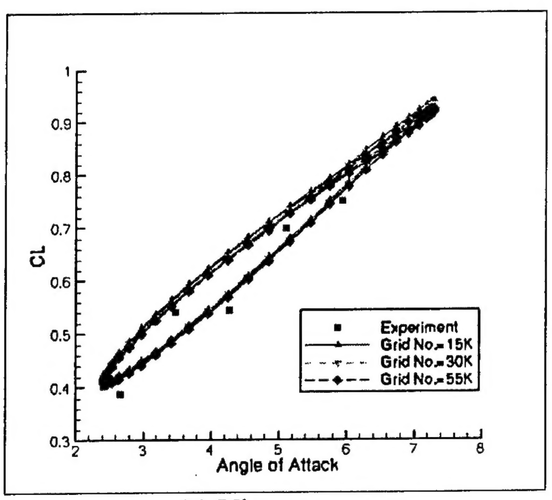
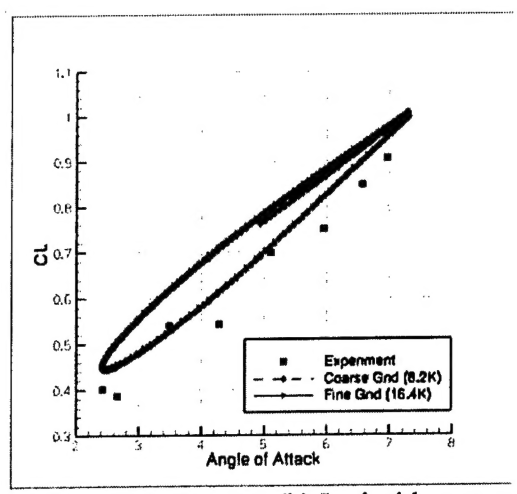


Figure 4.5. Lift coefficient vs. angle of attack for both inviscid and viscous simulations



(a) Viscous



(b) Inviscid

Figure 4.6. Lift coefficient vs. angle of attack computed on different grids

5. CONCLUSIONS AND FUTURE WORK

Since the award of this grant in the March of 2003, significant progresses have been made in the project. The overset adaptive Cartesian/prism grid generator has been completed. We are in the process of developing the hole cutting module to generate interpolation stencils for the flow solver MUSIC. In addition, several enhancements have been made to the flow solver to support the current project. They include the extension of the flow solver to unsteady moving boundary problems, and the implementation of a Spalart-Allmaras model, and a DES model. These implementations have been validated with several test cases. We are now testing the DES implementation.

We will complete the hole-cutting module, and implement a new interpolation boundary condition in MUSIC. Then we will verify the implementation with a demonstration case using an overset adaptive Cartesian/prism grid.

Based on the progresses we have made so far, we anticipate performing the following tasks in the future:

1. Complete the development necessary to run unsteady 3D moving boundary problems;
2. Parallelize the overset flow solver to perform large scale 3D computations;
3. Perform solution based grid adaptation in a time accurate manner;
4. Simulate moving boundary flow problems with "real world" geometry and flow conditions and compare with experimental data;
5. Deliver a versatile simulation environment to the Air Force for moving boundary flow problems with necessary documentations.

6. REFERENCES

1. Meakin, R. and Suhs, N., "Unsteady Aerodynamic Simulation of Multiple Bodies in Relative Motion," in Proceedings of AIAA 9th Computational Fluid Dynamics Conference, Buffalo, NY, pp. 643-657, June, 1989.
2. Meakin, R., "An Efficient Means of Adaptive Refinement Within Systems of Overset Grids," in Proceedings of AIAA 12th Computational Fluid Dynamics Conference, San Diego, CA, pp. 844-857, June, 1995.
3. Meakin, R., "On Adaptive Refinement and Overset Structured Grid," in Proceedings of AIAA 13th Computational Fluid Dynamics Conference, Snowmass, CO, pp. 236-249, June, 1997.
4. Meakin, R. and Wissink, A.M., "Unsteady Aerodynamic Simulation of Static and Moving Bodies Using Scalable Computers," in Proceedings of AIAA 13th Computational Fluid Dynamics Conference, Norfolk, VA, pp. 469-483, June, 1997.
5. Lohner, R., "Three-Dimensional Fluid-Structure Interaction Using a Finite Element Solver and Adaptive Remeshing," Computer Systems in Engineering 1, 2-4, 257-272, 1990.
6. Lohner, R. and Baum, J.D., "Adaptive H-Refinement on 3D Unstructured Grids for Transient Problems," International J. Numerical Methods Fluids, 14, pp. 1407-1419, 1992.
7. Baum, J.D., Lohner, R., Marquette, T.J., Luo, H., "Numerical Simulation of Aircraft Canopy Trajectory," AIAA-97-1885, 1997.
8. Sharov, D., Luo, H., Baum, J.D. and Lohner, R., "Implementation of Unstructured Grid GMRES+LU-SGS Method on Shared-Memory, Cache-Based Parallel Computers," AIAA-2000-0927, 2000.
9. Luo, H., Sharov, D., Baum, J.D, and Lohner, R., "On the Computation of Compressible Turbulent Flows on Unstructured Grids," AIAA-2000-0926, 2000.
10. Wang, Z.J. and Bayyuk, S.A., "An Automated Adaptive, Unstructured, Cartesian-Prism-Based Technique for Moving-Boundary Simulations," in Proceedings of 6th International Conference on Numerical Grid Generation in Computational Field Simulations, London, July 1998.
11. Barth, T.J., and Frederickson, P.O., "Higher–Order Solution of the Euler Equations on Unstructured Grids Using Quadratic Reconstruction," AIAA Paper 90–0013, 1990.

12. Wang, Z.J. and Chen, R.F. "Anisotropic Solution-Adaptive Viscous Cartesian Grid Method for Turbulent Flow Simulation," *AIAA Journal*, Vol. 40, pp. 1969-1978, 2002.
13. Shur, M., Spalart, P.R., Strelets, M. and Travin, A., "Detached Eddy Simulation of an Airfoil at High Angle of Attack," 4th Int. Symp. Eng. Turb. Modeling and Measurements, Corsica, 1999.
14. Spalart, P.R., Jou, W.H., Strelets, M. and Allmaras, S.R., "Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach," First AFOSR International Conference on DNS/LES, Ruston, LA, 1997.
15. Forsythe, J.R., Squires K.D., Wurtzler, K.E. and Spalart, P.R., "Detached-Eddy Simulation of Fighter Aircraft at High Alpha," AIAA Paper No. 2002-0591.
16. Constantinescu, G.S. and Squires, K., "LES and DES Investigations of Turbulent Flow Over a Sphere," AIAA Paper No. 2000-0540, 2000.
17. Wang, Z.J. and Srinivasan, K. "An Adaptive Cartesian Grid Generation Method for 'Dirty' Geometry," *International Journal for Numerical Methods in Fluids*, Vol. 39, pp. 703-717, 2002.
18. Wang, Z.J. and Chen, R.F., "Parallel Adaptive Cartesian Grid Method for Viscous Flow with Complex Geometries," *CFD Journal*, Vol. 10, No. 4, pp.575-584, 2002.
19. Tomaro, R. F., Witzeman, F. C., Strang, W. Z., "Simulation of Store Separation for the F/A-18C Using $Cobalt_{60}$," Journal of Aircraft, Vol. 37, No. 3, May-June 2000, pp. 361-67.
20. DeZeeuw, D. and Powell, K., "An Adaptively Refined Cartesian Mesh Solver for the Euler Equations," AIAA-91-1542-CP.
21. Aftosmis M.J., Berger, M.J. and Melton J.E., "Robust and Efficient Cartesian Mesh Generation for Component-Based Geometry," AIAA Paper No. 97-0916.
22. Wang, Z.J. and V. Parthasarathy, "A Fully Automated Chimera Methodology for Multiple Moving Body Problems," *International Journal for Numerical Methods in Fluids*, Vol. 33, No. 7, pp 919-938, 2000.
23. Bonet, J. and Peraire, J., "An Alternating Digital Tree (ADT) Algorithm for 3D Geometric Search and Intersection Problems," International Journal for Numerical Methods in Engineering, Vol. 31, pp. 1-17, 1991.
24. P.R. Spalart and S.R. Allmaras, "A One-Equation Turbulence Model for Aerodynamic Flow," *La Rech. Aerospatiale*, Vol. 1, pp.5-21, 1994.